# Computational Ship Hydrodynamics for Revolutionary Naval Combatants

### *DoD Challenge Project: Time-Domain Computational Ship Hydrodynamics*

Edwin P. Rood
Office of Naval Research
Arlington VA 22217

## Abstract

A grand challenge for computational fluid dynamics is the modeling and simulation of the time evolution of the fully nonlinear turbulent free-surface flow around surface ships. This paper reports on a DoD Challenge Project team progress toward the project goals of predicting the turbulent hydrodynamic free-surface flow for the DDG-51 destroyer maneuvering in waves and for a generic concept representing the new class of land attack destroyer, the DD-21. The Challenge Project team uses four software applications, which are as follows with the lead team member in parens: CFDSHIP-IOWA (E. Paterson and F. Stern), UNCLE (J. Gorski, as developed by MSU), NFA (D. Dommermuth), and SHIPLES (D. Yue). These applications employ both Reynolds-Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) approaches to modeling the turbulent free surface flow around a combatant. All four applications predict the entire flow around the surface ship; yet, each application is designed to focus on particular features of the flow. Application of this suite of software codes provides mutually supportive information covering the range of scales from turbulence to ship maneuvering. The platforms used are the Origin 2000, Cray T3E, and IBM SP3.

This Challenge Project began in late 1996 with the goal of producing computational methods with high spatial and temporal resolution for surface ship combatant hydrodynamics for use in ship design procedures to meet signature requirements. The requirement has had a profound impact on applications of naval hydrodynamics, necessitating an expansion in focus from macroscale performance (e.g., drag and powering) to microscale flow feature definition. In 1996, whereas the equations of motion were understood, the ability to solve those equations was only beginning to be realized (Rood, 1996). Massive computational capability and the expertise to wisely use that capability have grown exponentially during the past couple of years (Ratcliffe, et al., 1998) such that, today, computations are frequently made for complex ship geometries, such as integrated propulsor/hull concepts for advanced naval combatants. As a result, new procedures are forthcoming for design analysis for hydrodynamic signatures, and the computational methods are being used in the acquisition process for the Navy's new land attack destroyer, the DD-21.

## The Challenge

The challenges for this project include both the computational solution for the complex hydrodynamics for a surface combatant, and the efficient implementation of the solution process on supercomputers so that the software is useable for design evaluation.

Naval hydrodynamic design procedure for new ships traditionally relies on regression analysis and towing tank testing using data bases developed from decades of model test results correlated with full scale ship experience. New combatant concepts, examples of which are shown in Figure 1, are sufficiently revolutionary that existing databases do not provide the necessary design information. This deficiency becomes even more critical in the light of increased demands for stealthy operation for future combatants. Furthermore, the time required to develop a database using existing "build and test" procedures is not practical. The Navy is therefore developing a new paradigm for hull and propeller design by recognizing the crucial need for computational methods to greatly enhance traditional towing tank model testing. The computational need is to complement towing tank experiments with numerical towing tank investigations that 1) rapidly expand the database, 2) extend the resolution of physical measurements, and 3) provide an economic foundation for parametric exploration.

For some time the Navy has invested in computational techniques for submarines. The aircraft industry has for some time invested in computational methods to predict lift on wings and bodies. The challenge in naval hydrodynamics, compared to aerodynamics, has been the incompressibility of water and the need to predict the drag on bodies as well as the lift. The additional challenge for surface combatants is the need to solve for the deformations of free surface, such as waves and turbulent fluctuations, in addition to the pressures and velocities of the flow around the ship. These challenges place this Challenge Project at the frontier of both basic and applied fluid dynamics.

The technical challenge undertaken by this project requires the solution to unique and fundamental problems in fluid mechanics. These include the understanding, modeling and simulation of free-surface turbulence, free-surface and body juncture turbulent flow involving dynamic contact lines, unsteady flow separation and moving separation lines, and breaking waves and spray. The computational requirements pose a unique challenge to large-scale computations, including scalability and efficiency for a parallelized approach to solving the problem (Rood, 1998b).

The computational challenge is to solve for a large number of time steps and a large number of spatial grid locations in an efficient manner. The need to solve for the location of the water surface boundary along with the flow pressures and velocities requires a large number of time steps, for each of which the entire flow domain must be computed. Additional time steps are required to iterate the effects of the propulsor in the interaction between the propulsor and the hull, including free surface deformations. The need to discern the dominating effects of the shear produced by velocity gradients in the flow necessitates dense grids. An additional gridding challenge is to accurately resolve complex geometries, including highly curved hull surfaces and intersections of propulsor support structure with the hull. To be of use in design and evaluation of a set of concepts, the computations must complete within one day of building the grid representing the ship geometry and the flow domain, including the free surface deformations.

This DoD Challenge Project has focused massive resources to develop the first-ever computations for the turbulent flow around a naval combatant. This development by application is in the context of a rigorous verification, validation, and calibration methodology upon which well-founded design and analysis procedures are under development. As an expansion to the original Challenge Project, the application to integrated propulsor/hull configurations relevant to the DD-21 land attack destroyer acquisition program is a priority. This brought additional challenges in terms of transitioning the working software to industry, drawing in industrial computational experts as part of the Challenge project to effect successful transition. The pace of revolution, from academic software to industrially verified and validated applications in three years, has been fast. The goal was achieved, and surface combatant hydrodynamic design and analysis procedures have been forever changed.

## Approach and HPCMP Support

Application of the computational methods to the time dependent flow around the DDG-51 rapidly improved the quality and usability of computational results as new HPCMP resources became available. This project was initiated with specific computers in mind, which generally mirrored the serial computers used to originally develop the codes. Development and application of the parallelized versions of the code have occurred at the same time that advanced resources have been made available. The reader is referred to previous papers (Rood, 1997; Rood, 1998a, Rood, 1999) for details on the evolution of the resources used in this project. The following are updated descriptions of the software.

CFDSHIP-IOWA version 3.2 (Paterson et al., 2000) is a general-purpose research code that solves the unsteady RANS equations in either time-accurate or steady-flow modes. The basic flow solver uses structured multiblock grids, the PISO algorithm, vertex-centered collocated flow variables, and higher-order finite differences. The data structure permits either serial or parallel execution, provides a high-level of portability, and eases subroutine development (e.g., turbulence, two-phase, stratification models) by the user community. A NAVO PET Tiger Team collaboration (Paterson and Sinkovits, 1999) successfully accelerated code development. A coarse-grain parallel distributed-memory multi-block approach is used where grid blocks are mapped 1:1 with processors. Memory utilization scales linearly given uniform distribution of block sizes. Challenge Project work (Paterson et al., 2000; Paterson and Sinkovits, 1999) has shown that CFDSHIP-IOWA attains linear acceleration with up to 64 processors, on both the SGI Origin 2000 and the Cray T3E. Portability across the available HPCMO platforms is achieved using UNIX make utility, Message Passing Interface (MPI), and C pre-processor statements for isolation of parallel and platform-specific coding. However, the target platform for code development has been the Origin 2000 since cache optimization was performed for this machine prior to parallel-code development. In addition, the symmetric multi-processor (SMP) architecture promises to lend itself to hybrid multi-level parallelism using both MPI and OpenMP. Also, it should be noted that CFDSHIP-IOWA efficiently runs on the Cray T90 (i.e., approx. 350 MFLOPS) and SV1, which have been heavily used during the IIHR group training and transition to the Origin.

Problems encountered in application of CFDSHIP-IOWA are primarily due to grid generation, domain decomposition, and load balancing. Unfortunately, even with commercial software, generation of block-structured grids for practical geometries is difficult. Complicated geometric features (sonar dome, transom stern, propulsor, and other appendages), dynamic free-surface conforming grid, and clustering to resolve important physics (e.g., near-wall boundary layers and off-body wave fields) requires large number of blocks of disparate size. In contrast, coarse-grain parallel multi-block requires that the grid system be partitioned into blocks of equal size for optimum performance. Until code development can be completed for dynamic and automatic load balancing (i.e., hybrid MPI/OpenMP multi-level parallelism), static load balancing is used, i.e., burden is placed on the user generating the grid to decompose the initial grid system into blocks of approximately the same size. This can be accomplished using block manipulation tools in (i.e., GRIDGEN), however, it still requires substantial time by an expert (thus the need for serial T90 time for use by junior investigators).

The UNCLE code solves the three-dimensional unsteady incompressible RANS equations in transformed coordinates using dynamic relative-motion multiblock grids. The basic solution algorithm is discretized as a cell-centered finite-volume approximation, and the cell-face numerical flux vectors are represented by third-order high-resolution Roe/MUSCL flux approximations. For unsteady flows, the discrete approximation for the complete nonlinear unsteady incompressible equations is solved iteratively using an artificial compressibility formulation. For steady flows, the artificial compressibility formulation is solved directly using a closely related pseudo-time iteration algorithm. The flux derivatives required for linearization of the high-resolution Roe fluxes are not easily obtained analytically, so these flux derivatives are computed numerically using the state-variable linearization technique.

The scalable parallel algorithm in UNCLE combines domain decomposition and message passing for concurrent solution, with an iteration hierarchy combining Newton or time linearization, full-approximation multigrid acceleration, and concurrent block-Jacobi/Symmetric Gauss-Seidel for the innermost iteration. This algorithm involves point-to-point message exchanges at each subiteration level. The code uses MPI because of its extensive portability and functionality. Static load balancing is done either at the grid-generation stage or using a graphical repartitioning tool developed for preprocessing of existing grids. A heuristic performance estimator which takes into account the characteristics of the algorithm and the available system resources guides the problem decomposition and case setup. The practical geometries of interest to date have been treated using multiblock dynamic structured grids connected in an arbitrary (unstructured) pattern, including the possibility that a block surface can be subdivided to connect with more than one adjacent grid block. The dynamic grid has capabilities for movement of the overall grid, local grid deformation and relative motion between grid blocks. The grids require point to point connectivity that can put some strain on grid generation. This can be alleviated somewhat with the overlaid grid capability available in CFDSHIP-IOWA.

Both portability and scalability of the flow solver is important. For the UNCLE codes, software portability is addressed through the use of MPI and FORTRAN 90. The existing structured code has been demonstrated for numerous parallel platforms including T3E, IBM SP-2, Sun Enterprise ULTRA 10000, SGI-O2K and PCA Arrays, as well as workstation clusters. The parallel solutions typically have communications overhead of only 10-15%. Scalability studies using heuristic performance estimates indicate that on current-generation hardware, parallel efficiencies (percent CPU utilization) of 80 percent and more can be achieved on up to 400 processors and 50 million points, using appropriately sized grids.

Performance of the UNCLE solver can best be described by looking at some examples. Of interest in the current study is the capability to simulate hull flows with nonlinear free-surface effects. The run time for a DDG-51 hull for a very difficult case involving a transom stern, unsteady wake, and wetted stern is about 27 hours on 32 processors of a T3E, for 3500 time steps using 2.07 million points. Simulations for a notional single-stage complex propulsor requires 20 hours on 87 T3E processors for 1000 time steps using a grid of approximately 11 million points. The code runs at about 72 MFLOPS per processor, excluding communications overhead. This case requires 22.3 Gb of memory and runs at a net rate of about 4.2 GFLOPS, including the effects of a 20% load imbalance and 15% communications overhead.

Application of the UNCLE code to the DD-21 concept geometry involved three grids of 1.75 thousand, 1.25 million, and 9.5 million points for the verification and validation process. The coarse grids were obtained by removing every other point in all three directions from the next finer grid. Computations for all three grids were performed on an Origin 2000 using 28 processors. The coarse grid solution could be run for 10,000 cycles in less than ½ day of CPU time. The real clock time was equivalent to the CPU time once the job started running. However, the coarse grid solutions did not represent the free surface very well and are probably too coarse to be used as a design tool. The same 10,000 cycles took about 4 days of clock time for the medium grid. The medium grid provides very reasonable predictions of the flow field and can be used to make design decisions. The 10,000 cycles was to ensure the solution was absolutely converged for the verification and validation process. However, for practical purposes the solution was often converged within 5,000 cycles or 2 days once the job started executing. These are walk clock times starting a run from scratch without a good guess of the initial conditions. In a design cycle where one is making modest changes to the geometry one would typically start the calculation from the solution for a previous geometry. Calculations of this sort would often take about ½ as much time again getting a converged solution within one day. One could also use more processors if they are available easily fulfilling the requirement mentioned earlier of needing solutions within one day for a practical design tool. Running the 9.5 million point grid on the same 28 processors for 10,000 cycles took about a month of real time. This also included wait times while the job was not executing between runs. Although this calculation provided some very high resolution of the flow field one would only want such large scale computations done to evaluate a final design and not part of design cycle where the geometry is changing. Once again however savings can be obtained by running only 5,000 cycles and using more processors to get the time down to a reasonable amount.

The NFA code uses a two-phase LES model of the Navier-Stokes equations to simulate spray and turbulent wakes. NFA is written in High Performance Fortran (HPF), and is currently running on the NAVO T3E. The core solver of NFA uses multigrid and line Jacobi iteration to solve a Poisson equation that has variable coefficients. The CPU times for various grid sizes are provided in Table 1.

| Grid Points | Cray T3E Nodes | | | | |
|---|---|---|---|---|---|
| | 8 | 16 | 32 | 64 | 128 |
| 262,144 | 7.7 | 5.0 | 3.8 | 3.8 | 5.8 |
| 2,097,152 | -- | 25.8 | 14.7 | 9.4 | 9.2 |
| 16,777,216 | -- | -- | 96.4 | 50.8 | 30.7 |

Table 1. CPU seconds required to solve Poisson's equation.

Three different grid sizes are considered: $64^3$, $128^3$, and $256^3$ grid points. The CPU times are based on solving a Poisson equation with variable coefficients to machine accuracy. As the number of CPU nodes increases for a fixed grid size, performance eventually degrades due to communication costs, but only for problems where the grid size is small compared to the number of nodes. For a fixed number of CPU nodes, the communication costs are significantly reduced (on a relative basis) as the number of grid points is increased. As a result, for 32 cpu nodes, it takes much less than eight times longer to solve the $128^3$ problem than the $64^3$ problem. A significant finding is that the amount of memory may be more important than the number of processors for achieving very large computations. Improvements to the core solver are currently being pursued in collaboration with a Tiger team located at the University of California at San Diego. Some of this work is the implementation of key parts of NFA in MPI. The work on the tridiagonal solver is now complete. The new MPI tridiagonal solver is about four times faster than the original HPF tridiagonal solver. The next upgrade is focusing on key aspects of the multigrid algorithm.

Recent upgrades to NFA include a new procedure for initializing simulations of spray sheets and turbulent wakes. Numerical simulations have been performed which give new insight into the breakup of spray sheets. In particular, the shedding of vorticity within the air has been captured with high fidelity. The shedding thickens the boundary layer. The numerical capability for simulating wakes using LES is nearing maturity. For submerged wakes, excellent comparisons to laboratory data have been performed. These comparisons were made possible by a new procedure for initializing LES of ship and submarine wakes. The new numerical procedure establishes the correct balance between turbulent production and turbulent dissipation. This is a significant breakthrough because earlier simulations did not establish the correct balance and as a result, decayed too quickly. The new numerical results show that stratification leads to the formation of pancake eddies in the far wake of submerged bodies. The pancake eddies cause the wake to persist longer than it would in the absence of stratification, which agreed very well with laboratory data. Similar effects are expected for surface ships, although the presence of the free surface may lead to new interactions.

The grid construction method of NFA has upgraded to allow variable mesh spacing. In addition, recent upgrades to the Portland Group HPF compiler made it possible to rewrite the multigrid solver using recursive subroutine calls, which greatly simplified the source code. As a result, NFA will be much easier to maintain. Recently the NFA method was upgraded in capability by the addition of both level set and volume of fluid methods to capture breaking waves and air entrainment. The new upgrades permit better modeling of the breaking wave at the bow and the rooster tail in the stern (Sussman & Dommermuth, 2000).

The SHIPLES code is used on the IBM-SP3 (ASC) to take advantage of the single node computational speed and the highly scalable distributed memory architecture. Computations on the IBM-SP3 permit the usage of a large number of nodes and allow for memory per node of O(1GB). The SHIPLES program uses a coarse-grained data decomposition methodology to obtain highly optimized algorithms. Parallelization is achieved via a FORTRAN parallelization utility, xHPF (Applied Parallel Research, Inc.), with additional MPI extrinsics for an iterative SOR Poisson solver and an array transpose routine coupled into the algorithm. This type of methodology, coupling xHPF and MPI extrinsics, accounts for a large portion of the increased efficiency seen in SHIPLES. Significant improvements in per node memory utilization and body geometry representation further allow SHIPLES to run high resolutions on a smaller number of nodes, increasing the highest realizable resolution for SHIPLES on available DoD HPCMP platforms.

Typical computer resources required for the software applications are provided in the following table:

| Code | Computer | Processors | Memory | Communications | Note |
|---|---|---|---|---|---|
| CFDSHIP-IOWA | Origin 2000 | 1-128 | 2GB-5GB | MPI | |
| UNCLE | Origin 2000 | 28 | 23GB | MPI | |
| NFA | Cray T3E | 64-256 | 300MB/node | HPF | Portions to MPI |
| SHIPLES | IBM SP3 | 16-32 | 1000MB/node | XHPF/MPI | MPI Extrinsics |

## Solutions for DDG-51

The evolution of the computational capabilities under this Challenge Project for the DDG51, a conventionl transom-sterned Navy destroyer, is documented in previous publications (Rood, 1997, 1998a, 1998b, 1999). The computational focus for this paper is on 1) validating the RANS solutions for the DDG-51 in ambient waves, and extending the computational capability to include unsteady hydrodynamics produced by ship pitching and heaving motions, 2) restructuring the NFA software to produce a hybrid volume of fluid, level set, large eddy simulation capability for breaking waves and air entrainment, and 3) continuing the validation of mixed turbulence modeling large eddy simulation for bow flows and ship wakes.

Figure 2 shows the axial velocity and free-surface contours at one instant in time and the unsteady drag for the DDG-51 operating in head seas. The solid line is the prediction of the drag, and the circle symbols are the corresponding experimental measurements. The amplitudes of the data are within 10 percent, and the phase within 30 degrees. It is believed that the differences are produced by the details in the physics of the flow at the transom stern, which for the Froude number for this flow is partially wetted. Partially wetted transom stern wakes are similar to bluff body wakes, although the deformations of the free surface are important and the flow is characterized by three dimensional effects. The time-accurate simulation of the DDG-51 in regular head seas required roughly 780,000 total grid points to accurately resolve both the turbulent boundary layer and incident waves on one side of the ship (i.e., a half-domain simulation was performed). Simulation of 5 periods of the incident wave required 22 hours wall clock time using 20 processors of the Origin 2000 and 210MW total memory.

The unsteady RANS solution has been extended to include motions of the ship itself, and a representative solution is shown in Figure 3. Figure 3 is an instant in the hydrodynamic and ship hull motion for the Wigley hull pitching and heaving in calm water. The computations show the growth and decay of the turbulent boundary layer along the hull, and into the propeller plane. Because the Wigley hull geometry is analytically defined and relatively simple compared to that of the DDG-51, this simulation was performed as a preliminary test of the grid conforming and ship movement algorithms. Planned simulations for the DDG-51 with ship motions will be considerably more complex and computationally demanding and will require full utilization of HPCMP resources. Estimates based on simulations of the DDG-51 in head seas and Wigley hull with ship motions indicate that roughly 1.6 million grid points and 1000 CPU hours (25 hours wall clock time) will be required to simulate both sides of the DDG-51 in pitching and heaving motion on the Origin 2000.

In both the computation and the physical flow, the vorticity and the free surface are fluctuating in time on account of the passage of large scale flow structures, or “whirls”. Whereas the RANS computations produce the average flow as determined by a time average longer than the turbulence time scales, and in some cases an unsteady flow such as that produced by rolling or turning, the LES computations capture the detailed spatial and temporal evolution of the locally unsteady flow produced by turbulence. The objective of this project is to produce consistent RANS and LES computations that link the turbulent eddies to the momentum driven flow modeled by the time average flow. The development of longitudinal vortices produced by the lift and three-dimensional flow separations on the bow dome were investigated with the LES application SHIPLES. The results for Figures 4 and 5, were generated using 16 IBM-SP3 nodes with O(1GB) memory per node and 21 days of CPU time. Figure 4 shows a three-way comparison between results obtained from experimental measurements (INSEAN), RANS (CFDSHIP-IOWA) and LES (SHIPLES) of the DDG-51 with no ambient waves for a particular transverse cut along the hull (x/L=0.1). The experimental measurements and RANS data are both time-averaged results while the LES is a single instant in time in the start up from the ship at rest to a steady speed. Seen in the figure are color contours of transverse velocity (V), velocity vectors (black) and streamlines (white). Overall comparison among the three is qualitatively and quantitatively excellent with the LES result capturing the structure near the bulb that is somewhat discernable from the experimental data. There are, however, differences that can be distinguished between the time-averaged and instantaneous data: (1) the free surface in the LES image is still moving as the bow wave is propagated away from the hull resulting in differing velocity profiles and causing the streamlines to turn towards the free-surface; and (2) a structural difference in transverse velocity contours exists near the bulb which is associated with transient vortical structures convected along the hull.

Figure 5 plots select data for the same LES results highlighting some of the unsteady features associated with the developing flow. Group "a" shows the time history of the same transverse cut shown in Figure 4. The time sequence shows the rise and fall of the free surface and the resulting shed vorticity as well as the quasi-steady structures near the bulb. Group "b" contains isosurfaces of longitudinal vorticity which show the three-dimensional vortical structures (note that isosurfaces of vorticity do not directly correspond to material vortex structures). As the free surface rises and falls along the hull during steady forward ship motion (Figure 5-b1), vortical structures form and are shed from the hull with the wave as it propagates away from the hull (Figure 5-a1 to Figure 5-a3). As with any free-surface body junction type flow, these structures are continuously shed in an oscillatory manner as the bow wave rises and falls. Unlike these unsteady structures in the bow wave (Figure 5-b3), fairly steady structures exist along the hull (near the bulb as seen in Figure 5-b2). While not shown here, when LES results are averaged over time, the resulting data compares well with RANS and averaged experimental measurements (as shown in Figure 4) yet loses the inherently unsteady flow features. The time accurate LES simulations of SHIPLES capture the steady and unsteady structures of the flow.

The addition of the level set and volume of fluid upgrades to the NFA code permit relatively accurate simulations of the breaking bow wave on the DDG-51, as shown in Figure 6. Surface ship signatures are determined in part by the flow disturbances left in the wake of the ship. As demonstrated in Figure 6, recent upgrades to the NFA code permit relatively accurate simulations of the breaking bow wave on the DDG-51. The solid line is a whisker-probe measurement. The green dot is a profile measurement. The red lines indicate the outline of the hull. The solid blue line is a level-set prediction. The small black dots are body-force points. Only half the body-force points are plotted. Whisker probes measure the free-surface elevation from the top down. The jump in the measurements occurs where the whisker probe came in contact with the overturning wave. The inserts provide perspective views of the laboratory experiments and the numerical simulations. The simulations were performed on the NAVO T3E using 64 CPU nodes with 25MB per processor. Each time step used 600 seconds and 100 time steps were used altogether to perform the implicit algorithm.

## Transition to DD-21 Acquisition

The success of this Challenge Project in demonstrating the usability of advanced computational methods to predict the turbulent free surface flow around a ship, including into the propeller plane, led to a Navy decision to fast-track the capability into the DD-21 acquisition program. The "ONR Surface Combatant Accelerated Hydrodynamics S&T Initiative" was formed to support a collaboration between the government and the two teams competing for the acquisition award. Under this initiative the software has been applied to a variety of concepts for the DD-21, the Navy's new land attack destroyer. The signature requirements for this ship mandate revolutionary changes such as integrated propulsor/hull systems, wave piercing bows, tumblehome hull forms. Examples of the kinds of geometry under consideration are shown in Figure 1.

The products of this project are being applied to design evaluation of generic concepts representing the DD-21. Free surface disturbances related to signatures are of special interest for the design of the DD-21. Unsteady RANS predictions have been used to suggest designs and to evaluate the propulsor inflow and the stern wake for generic geometries. Figure 7 shows the free surface deformations immediately downstream from the stern for one of the configurations studied. The computational solutions have been used extensively to perform parametric studies of propulsor/hull integration features, and are proven to be more cost effective than the traditional "build and test" using model scale ships in towing tanks.

The application of the software to revolutionary ships is part of the Documented Solution. Documented Solutions are formal archives of both the numerical verification and experimental validation process performed to aid in decisions as to the worth of the software for design analysis. This Challenge Project, in concert with advanced diagnostic procedures for model scale experiments, has for the first time established a rigorous paradigm by which complementary numerical and experimental investigations are used collaboratively to design and analyze practical hydrodynamic engineering problems. Documented Solutions are the first step in the accreditation process leading to the use of computational methods as primary engineering tools.

## Lessons Learned

In the few years that this Challenge Project has been in existence, the team has lifted computational ship hydrodynamics from the halls of academia and placed it among the tools for ship design and design evaluation. The use of RANS and LES methods for ship hydrodynamics remains in the hands of only a few qualified users because of the skill required to set up the computational domain and interpret the results. There are several important lessons or guidance items that researchers and ship designers should be aware of in using these sophisticated applications.

The success of this project was made possible by massive resources permitting a critical mass of researchers to work interactively to solve a Grand Challenge. With sufficient funding and ample access to computer resources, the team was able to accomplish more in a short period of time than would be expected if the same resources were provided over a much longer time.

The issues of geometry representation and grid construction remain impediments to the widespread use of these applications. The movement to unstructured grids will potentially help with grid construction. However, there is an important need for automation in the grid construction process. The manual adjustments that must now be made by experts in hydrodynamics severely limits the number of skilled users of the applications. Simulation of the shear flow into the propulsors, in the region where the propulsor structure blends into the hull, shows a clear need for geometric resolution of both the flow gradients and the surface geometry. Inadequate resolution of the intersections of the surfaces leads to unrealistic flow separations at the local scale which often produce computational instabilities.

It is well known that turbulent flow predictions are stymied by the lack of appropriate models for the turbulence. The problem is further complicated when the flow involves interacting and highly skewed boundary layers, such as those associated with ship geometries. For such flows it is often impossible to determine whether discrepancies in the prediction are due to the turbulence model or the grid itself. This is further complicated by the fact that the turbulence model and grid are dependent on each other. Because of the massive memory available with the supercomputers, leading to the ability to use dense grids to resolve gradients, it is possible to compute some truly grid independent solutions for some very complicated flow fields. This will provide, perhaps for the first time, some insight of what flow discrepancies are due to the grid and which are actually due to the turbulence model used. In addition, there should be less reliance on any corrections to the turbulence models to account for inadequate grid construction. A consequence is that there is even more reason to require advanced grid construction techniques to reduce the manual burden of fitting grids to the flow.

The notion of Documented Solutions, in which numerical and physical approaches are complementarily used to develop a solution with quantified accuracy is at the heart of gaining acceptance of the applications in the design community. With the Documented Solution, the user has a measured assurance that the applications are appropriate for the intended use.

## Where It Is All Going

This Challenge Project has revolutionalized traditional but antiquated ship hydrodynamics design and evaluation procedures from the well-established towing tank based methods to the modern world of computational methods. The Challenge Project met its goal, yet there are many years of engineering experience to be gained before computational ship hydrodynamics reaches its potential. The Project has established a new paradigm, which is being rapidly accepted as a standard tool for ship design and evaluation. This Challenge Project will officially close in the near future as scheduled. New challenge projects in computational ship hydrodynamics are envisioned for six degree of freedom maneuvering and for wave breaking. The validation of this numerical capability will provide design and analysis information for future ships, especially the revolutionary stealth combatants. Such computations will be used to provide a calibrated numerical towing tank, to interrogate sparse experimental databases and to extrapolate model-scale data to full scale for performance and signature design and analysis. Other efforts underway to apply optimization methods to the software will permit the automatic design of geometry to meet performance criteria.

## Acknowledgements

The DoD Challenge Project Team is indebted to many persons who provided significant advice, support, and collaboration. The Team would like to express appreciation especially to M. Sussman, K. Hendrickson, H. Haussling, R. Wilson, and to L. Davis.

## References

Paterson, E.G., and Sinkovits, R.S., “Performance, Scalability, and Portability of a MPI-based version of CFDSHIP-IOWA: Results of a NAVO PET Tiger-Team Collaboration,” *Proceedings of the 9th DoD HPCMP Users' Group Meeting*, Monterey, CA, 1999.

Paterson, E.G, Wilson, R., and Stern, F., "General Purpose Parallel Unsteady RANS for Ship Hydrodynamics", *Computers and Fluids*, Jun, 2000.

Ratcliffe, T., Bulgarelli, P., Coleman, H., Stern, F., and Whitfield, D., “Validation and Verification of Free Surface RANS Codes”, *Twenty-Second Symposium on Naval Hydrodynamics*, National Academy Press, 1998.

Rood, E.P., “Validation Strategy For ‘RANS’ Computational Ship Hydrodynamics”, *2nd International Conference on Hydrodynamics*, Hong Kong, 1996.

Rood, E.P., “Time-Domain Computational Ship Hydrodynamics”, *Proceedings of the DoD High Performance Computing Modernization Program Users Group Conference*, San Diego, 1997.

Rood, E.P., "Time-Domain Computational Ship Hydrodynamics: Features of the Flow Around the DDG 51", *Proceedings of the DoD High Performance Computing Modernization Program Users Group Conference*, Houston, 1998a.

Rood, E.P., "Transient and Periodic Turbulent Hydrodynamic Flows for Surface Ships: A DoD Challenge Project Milestone", AIAA Paper 98-2931, 1998b.

Rood, E.P., “Complementary RANS and LES Computations for DDG-51 and Transition to DD-21 Acquisition”, *Proceedings of the DoD High Performance Computing Modernization Program Users Group Conference*, Monterey, 1999.

Sussman, M., and Dommermuth, D.G., "The Numerical Simulation of Ship Waves Using Cartesian Grid Methods," *Twenty-Third Symposium on Naval Hydrodynamics*, 2000.

## Additional Challenge Project Publications


Chen, B. and Stern, F., " Computational Fluid Dynamics of Four-Quadrant Marine-Propulsor Flow," ASME Symposium on Advances in Numerical Modeling of Aerodynamics and Hydrodynamics in Turbomachinery," Washington D.C., 21-25 June 1998.

Dommermuth, D.G., Schlageter, E.A., Talcott, J.C., Novikov, E.A., & Wyatt, D.C., “Numerical Simulation of Bow Waves and Transom-Stern Flows”, *Twenty-Second Symposium on Naval Hydrodynamics*, National Academy Press, 1998.

Paterson, E.G., Kim, J., and Stern, F., “Unsteady RANS Simulation of an Integrated Marine Propulsor,” Proc. 7th International Conference on Numerical Ship Hydrodynamics, Nantes, France, 1999.

Paterson, E.G., Wilson, R.V., and Stern, F., “CFDSHIP-IOWA and Steady Flow RANS Simulation of DTMB Model 5415,” 1st Symposium on Marine Applications of Computational Fluid Dynamics, McLean, VA, 19-21 May 1998.

Rhee, S.H. and Stern, F., "Unsteady RANS Method for Surface Ship Boundary Layers and Wakes and Wave Fields," 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, 25-27 May 1998, pp. 67-84, IIHR Reprint #1328.

Stern, F., Wilson, R., Coleman, H., & Paterson, E., “Verification and Validation of CFD Simulations”, *Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference*, FEDSM99-6913, 1999.

Sreedhar, M. and Stern, F., "Non-Linear Eddy-Viscosity Turbulence Modeling for Solid/Free-Surface Juncture Boundary Layer and Wake," ASME Forum on Numerical Modeling of Free Surface and Interface Fluid Dynamics, Vancouver, Canada, June 1997, IIHR Reprint #1257.

Sreedhar, M. and Stern, F., "Investigation of Developing Juncture Flows using Large Eddy Simulation," First AFOSR International Conference on DNS/LES, Louisiana Tech University, Ruston, Louisiana, 4-8 August 1997, pp. 417-424, IIHR Reprint #1326.

Sreedhar, M. and Stern, F., "Prediction of Solid/Free-Surface Juncture Boundary Layer and Wake of a Surface-Piercing Flat Plate at Low Froude Number," ASME J. Fluids Eng., Vol. 120, June 1998, pp. 354-362, IIHR Reprint #1314.

Sreedhar, M. and Stern, F., "Large Eddy Simulation of Temporally Developing Juncture Flows," Int. J. Num. Meth. Fluids,Vol. 28, No. 1, 15 July 1998, pp. 47-72, IIHR Reprint #1323.

Wilson, R., Paterson, E., and Stern, F., "Unsteady RANS CFD Method for Naval Combatant in Waves," *Twenty-Second Symposium on Naval Hydrodynamics*, National Academy Press, 1998.

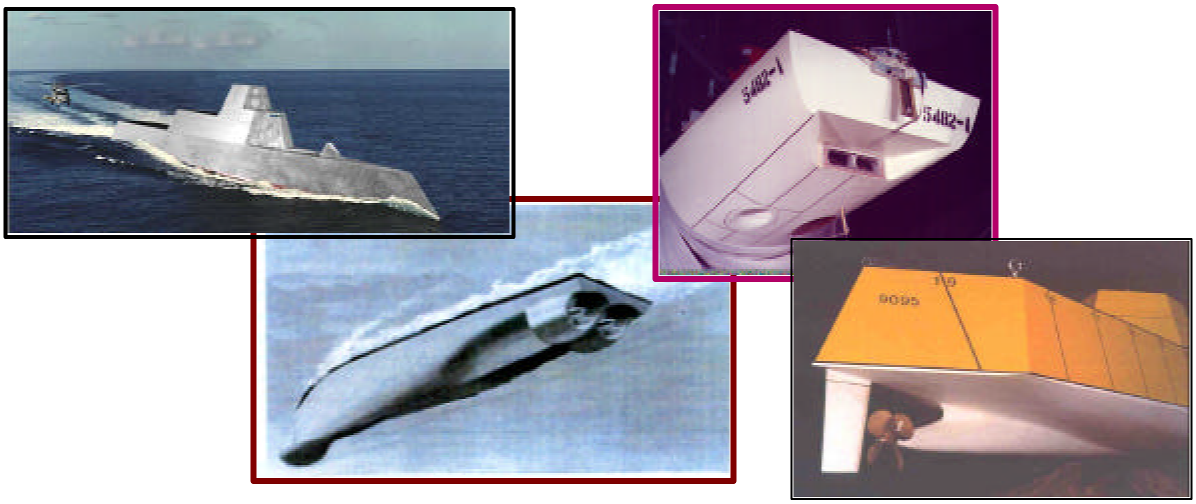


**Figure 1 - Representative Geometries for Revolutionary Surface Combatants**

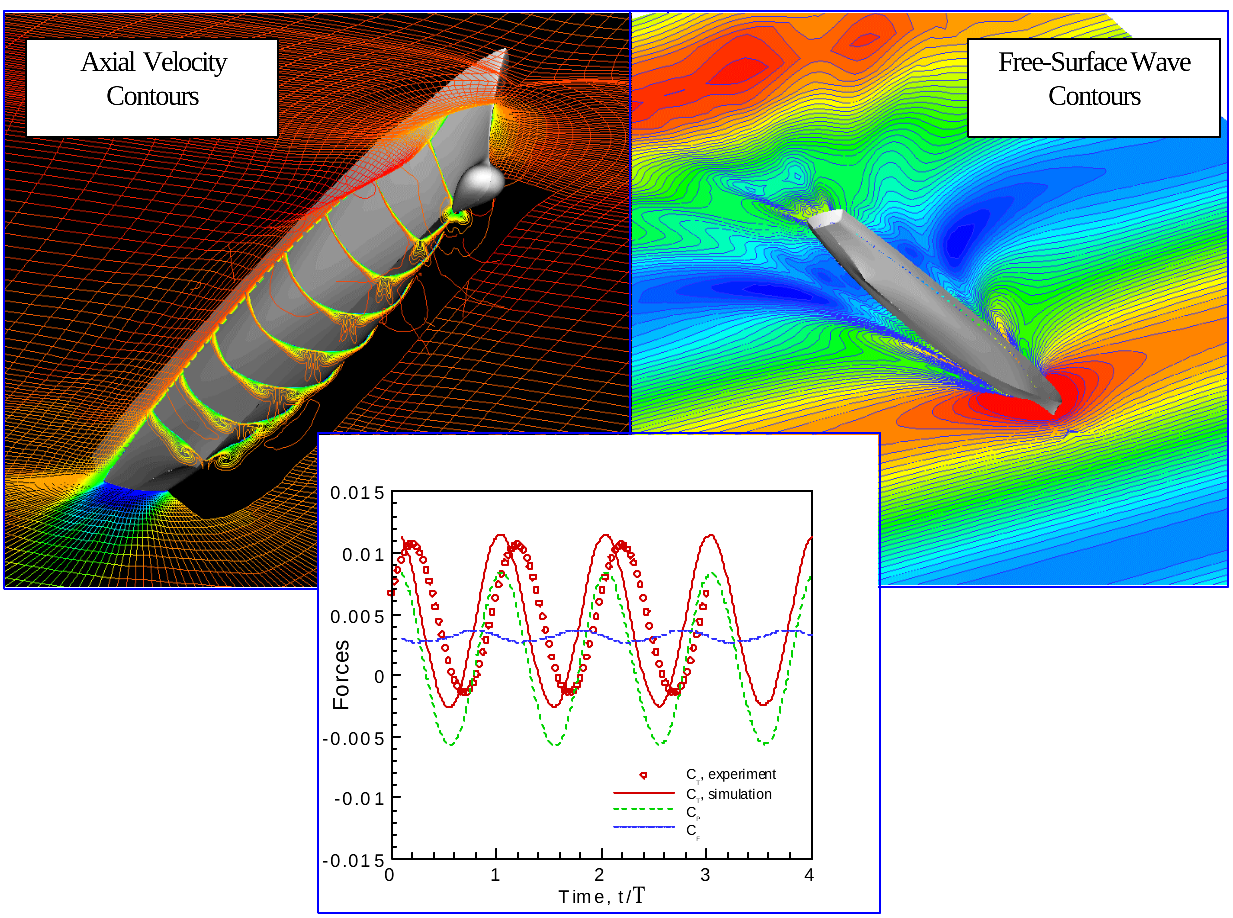


**Figure 2 – Unsteady RANS Prediction for DDG-51 in Regular Head Seas: Axial Velocity, Surface Contours, and Ship Forces (CFDSHIP-IOWA)**

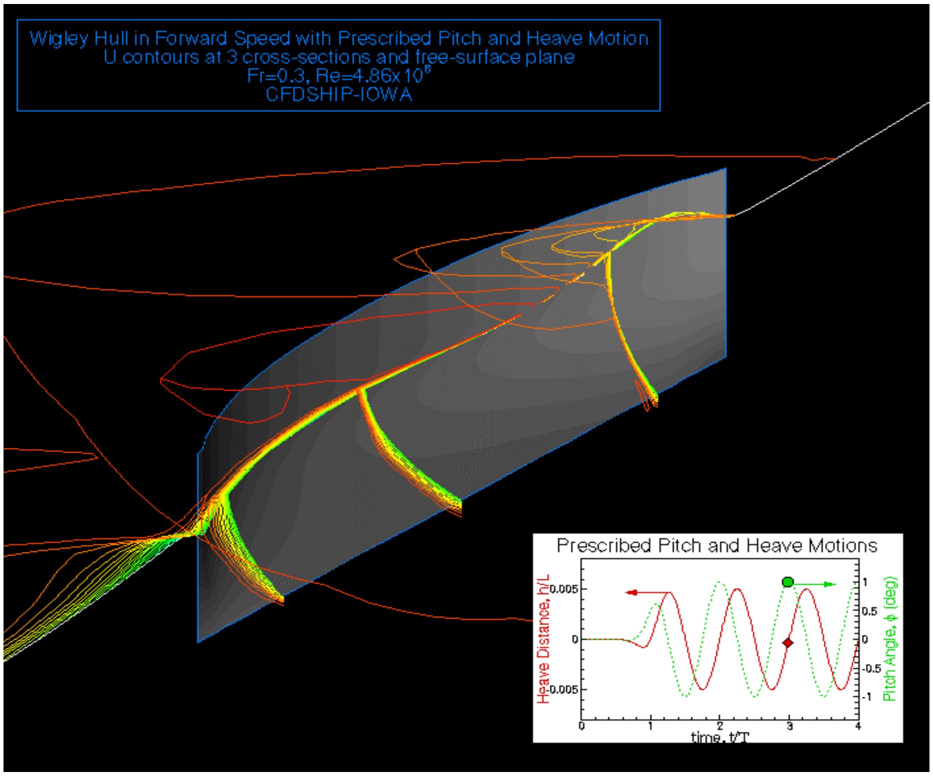


**Figure 3 - Boundary Layer Velocity Profiles Around the Wigley Hull Oscillating in Pitch and Heave**

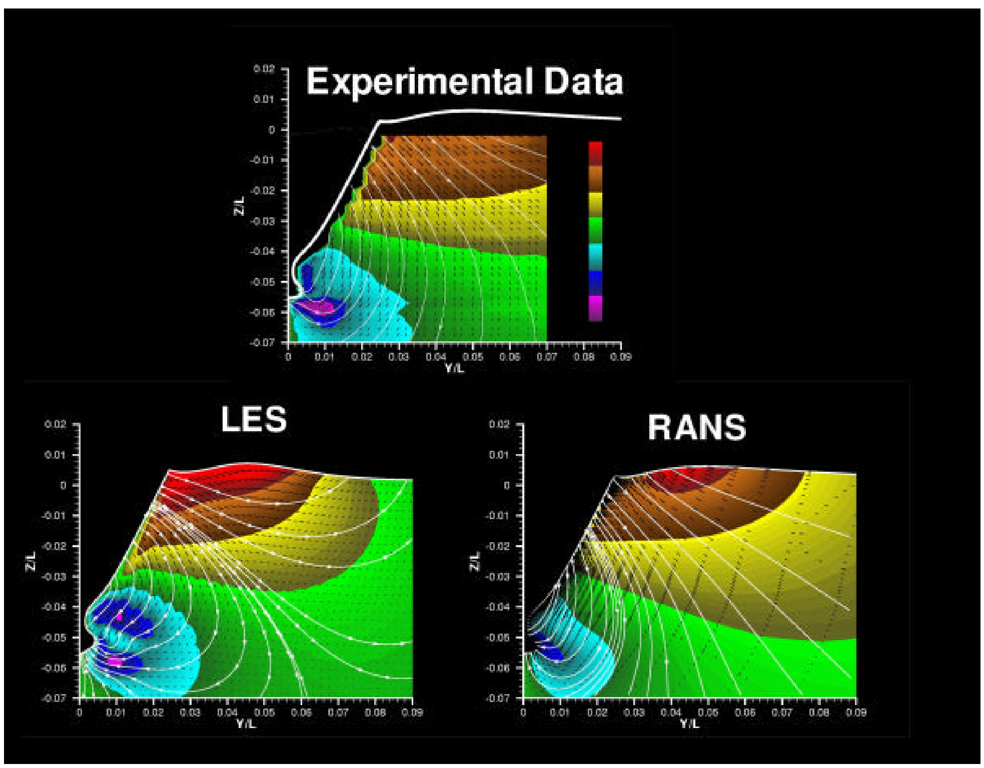


**Figure 4 – Comparisons of the Axial Velocities and the Streamlines for Experiment and RANS (Time Average Flow) and LES (Instantaneous Flow)**

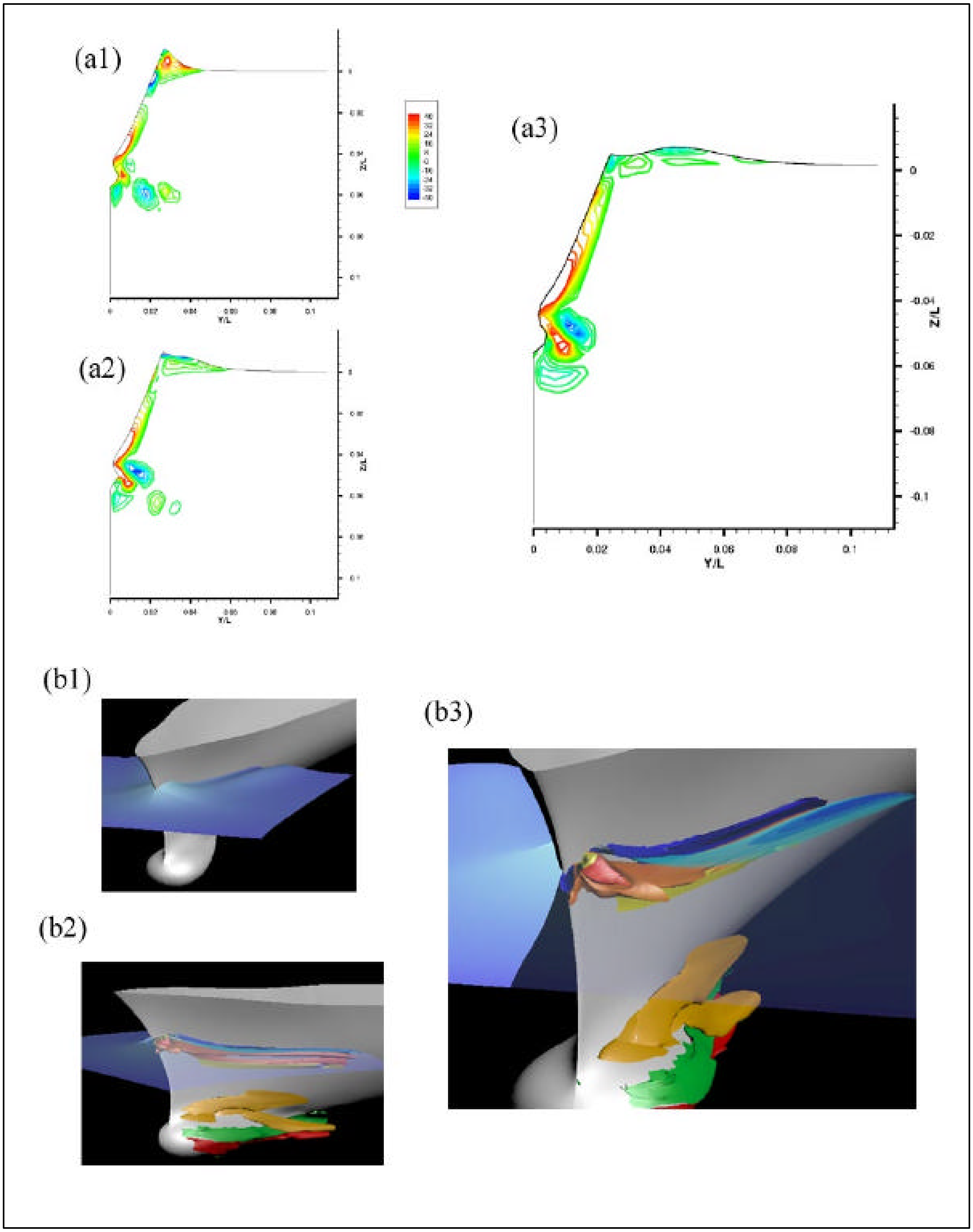


**Figure 5 – Development of the Longitudinal Vorticity Produced by the Sonar Dome Lift and Three-Dimensional Flow Separations (SHIPLES)**

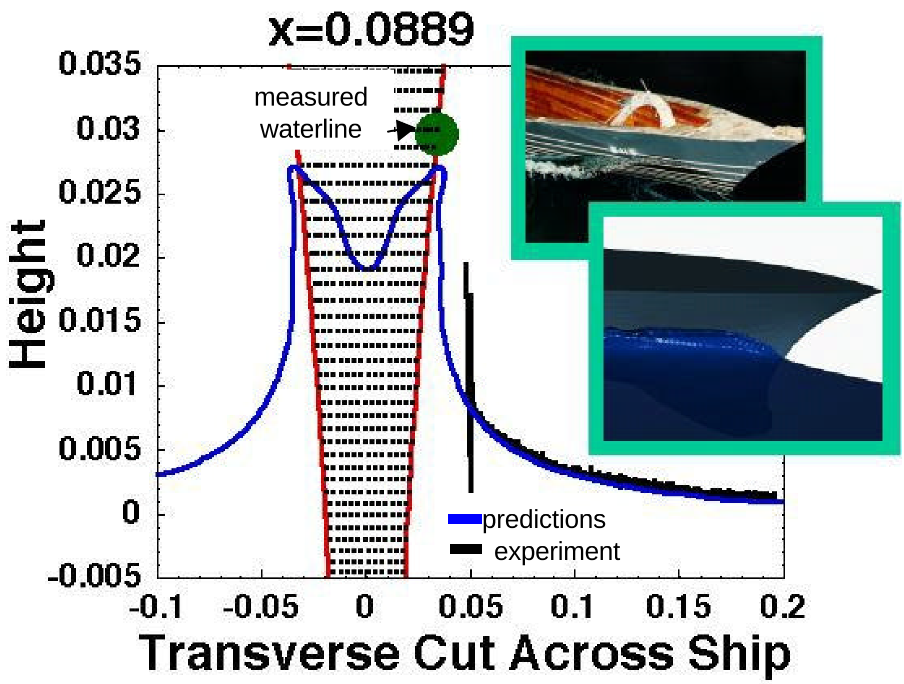


**Figure 6 – Coupled Level-Set and Volume of Fluid Method to Predict Time Evolution of Free Surface Elevations (NFA)**

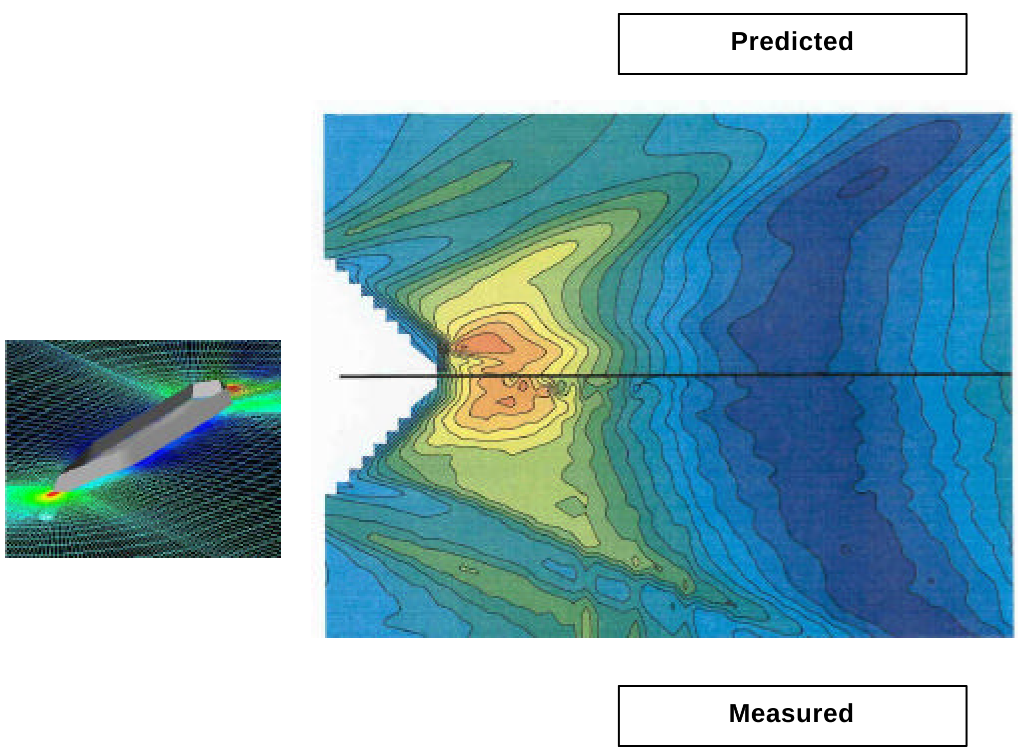


**Figure 7 – Unsteady RANS Predicted and Measured Stern Wave System for Generic DD-21 Propulsor/Hull (UNCLE)**